SAND-78-1407
Unlimited Release
Printed November 1978

COUPLEFLO
A COMPUTER PROGRAM FOR COUPLED CREEPING VISCOUS FLOW AND CONDUCTIVE-CONVECTIVE HEAT TRANSFER
PART II. USER'S MANUAL

P. F. Chavez and P. R. Dawson
Computational Physics & Mechanics Division I - 5531
Sandia Laboratories, Albuquerque, New Mexico 87185

ABSTRACT

COUPLEFLO is a two-dimensional finite element code for plane strain or axisymmetric analyses of thermomechanically coupled systems. It is capable of analyzing the creeping flow of non-Newtonian fluids or the secondary creep of solids. COUPLEFLO solves equations for conductive-convective heat transfer to determine the thermal response of a system. Thermomechanical coupling between the flow field and temperature distribution can exist in terms of temperature dependent material properties, temperature dependent body forces, viscous dissipation, material convection, and changing system geometry. Either transient or steady-state problems can be analyzed in Eulerian or quasi-Lagrangian reference frames. Part I - Theoretical Background contains the governing equation, finite element formulation, and verification of the code capabilities. Part II - User's Manual contains instructions for code use. Currently, COUPLEFLO is available at Sandia Laboratories in Albuquerque on the 7600, 6600, and NOS systems.

TABLE OF CONTENTS

LIST OF FIGURES

NOTATIONS

A	Area
$[B]$	Matrix coefficient modifier of
C_p	Specific heat
f	Any function
h	Convection coefficient
$[H]$	Matrix coefficient of
J	Functional
$[J]$	Jacobian matrix
$Det[J]$	Determinant of the Jacobian matrix
k, k_r, k_z	Thermal conductivity
$[K]$	Stiffness matrix
$[L]$	Surface heat transfer coefficient matrix
$[N]$	Shape functions
p	Pressure
q	Surface heat flux
Q	Heat generation rate
$[R]$	Conductivity matrix
s	Surface distance
S	Surface enclosing region V
S_q	Surface on which heat flux is specified
S_u	Surface on which velocity is specified
S_σ	Surface on which traction vectors are specified
S_θ	Surface on which temperature is specified
$\bar{T}$	Traction vector
t	Time

NOTATIONS (Cont'd)

Δt	Time increment
u, v	Velocity components
V	Volume of region
V_e	Volume of element
x_i	Cartesian coordinates
X	Body forces
δ	Variational operator
δ_{ij}	Kronecker delta
∇	Gradient operator
ε_{ij}	Strain-rate tensor
ε'_{ij}	Deviatoric strain-rate tensor
ε_{II}	Effective strain-rate $\left(\varepsilon_{II} = \left(\frac{2}{3}\varepsilon_{ij}\varepsilon_{ij}\right)^{1/2}\right)$
η, ξ, ζ	Local coordinates
λ	Lagrange multiplier
μ	Viscosity
ν_j	Unit outward normal vector
ρ	Density
σ_{ij}	Stress tensor
σ'_{ij}	Deviatoric stress tensor
σ'_{II}	Effective deviatoric stress $\left(\sigma'_{II} = \left(\frac{3}{2}\sigma'_{ij}\sigma'_{ij}\right)^{1/2}\right)$
θ	Temperature
$\dot{\theta}$	Time derivative of temperature
$\bar{\theta}$	Specified surface temperatures

Chapter I

INTRODUCTION

This document describes the input data and control cards necessary to use the computer code, COUPLEFLO. It also contains sample problems that illustrate application of COUPLEFLO. Description of the governing equations, finite element formulation, and solution algorithms are contained in a separate document [1].

The computer code, COUPLEFLO, is capable of analyzing plane strain or axisymmetric creeping viscous flow of temperature dependent non-Newtonian fluids or viscoplastic solids. The analyses apply to systems in which inertia can be assumed to be sufficiently small to be neglected and elastic response can be assumed small in comparison to viscoplastic behavior. Nonlinear behavior can be analyzed in COUPLEFLO using either variable stiffness or pseudo-force methods, including the secant modulus, tangent modulus and initial stress techniques.

Thermal analyses in COUPLEFLO are performed using a finite element formulation of the conductive-convective heat transfer equation. A Crank-Nicholson implicit finite difference scheme is utilized for time integrations of the temperature distribution. Temperature dependent thermal conductivity and convective boundary conditions are included in the formulation.

Coupling between the thermal and mechanical solutions may be simulated using COUPLEFLO. This coupling includes viscous dissipation, temperature dependent material properties, temperature dependent body forces, material convection and changes in geometry. Either control volume (Eulerian) or control mass (Lagrangian) reference frames may be specified. Both transient and steady-state analyses are possible.

Chapter II

SAMPLE PROBLEMS

2.1 Introduction

The purpose of this chapter is to demonstrate to the user the procedure in setting up problems for solution by COUPLEFLO. This will be accomplished by solving two problems that were formulated to reveal some of the more interesting options available in the code. Where input cards are displayed the reader should refer to Chapter IV which contains card descriptions and format specifications for the input. In addition, any self-consistent set of units may be used for input to COUPLEFLO. In regard to this, no units will be shown for either sample problem.

2.2 Compression of a Viscoplastic Tube

The first problem considered was an example of plane strain deformation of a viscoplastic tube made of an aluminum-like material. The actual thermomechanical system we wish to model is depicted in Fig. 2.1. We assume that symmetry exists about both x and y axes and hence the problem is only solved in the quarter plane. Furthermore, the thermal flux is specified to be zero across the exposed cross sections and convection boundary conditions are imposed along the inside and outside surfaces. A line force is applied along a line parallel to the y-axis. The material deformations are followed in time in a planar geometry. The assumed constitutive equation is linear and independent of temperature and has the form

$$\sigma'_{II} = 131700 \, \epsilon_{II} \; . \qquad (2.1)$$

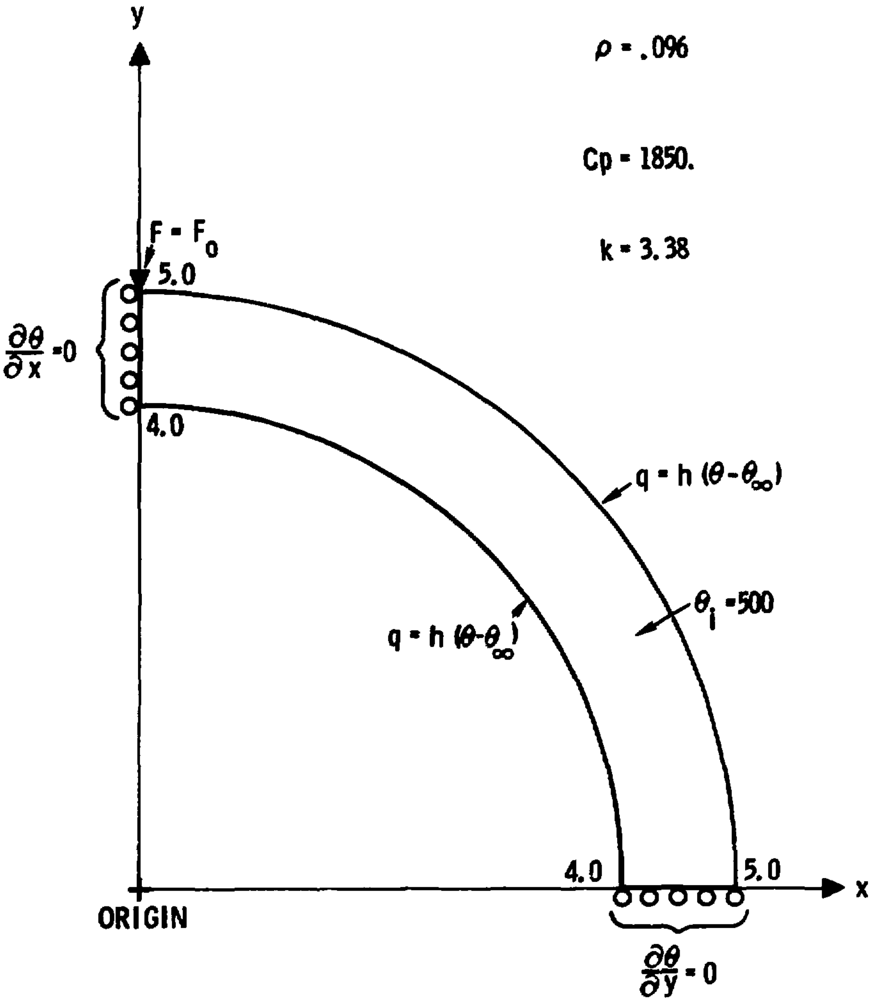
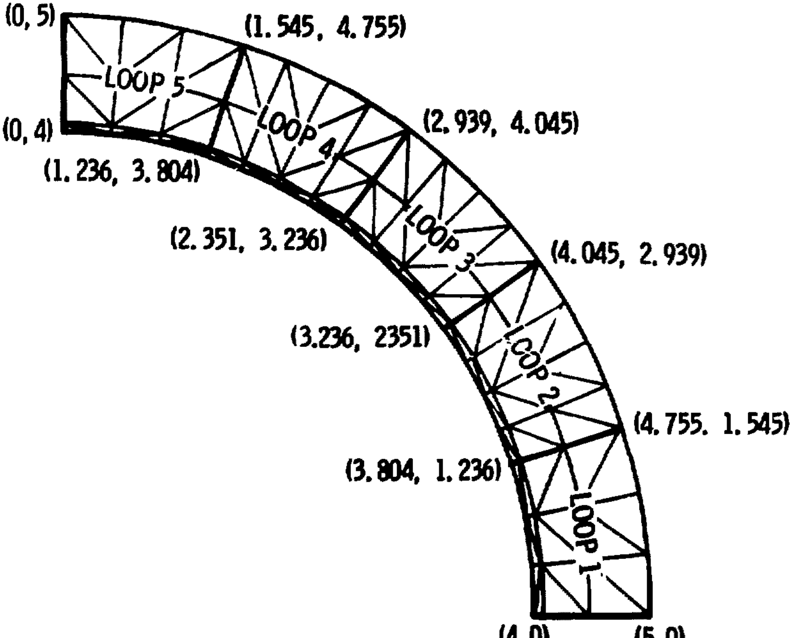


Figure 2.1. Plane Strain on a Viscoplastic Tube

Finally, the material composing the viscoplastic tube is assumed to have a density of .096, a heat capacity of 1350, and a constant conductivity of 3.38.

The solution of the problem is initiated by characterizing a quarter of the ring with triangular elements using MESH0. The following data set yields the quarter ring depicted in Fig. [illegible]

```
        5                                          Number of subregions

0.0      5.0      0.0      5.0                     plot limits

        1                                          slope of diagonal indicator

        9                                          write to TAPE9 nodal point
                                                      coordinates and element
                                                      definitions
        3        3
     0  0     0  0     0  0     0  0
4.0     0.0
4.25    0.0                                        Number of divisions, connec-
5.0     0.0                                        tion to previous subregions,
4.9384  0.7822                                     and subregion coordinates for:
4.7552  1.5451
4.0420  1.3133                                     subregion #1
3.8042  1.2361
3.9058  0.6257

        3        4
     1  3     0  0     0  0     0  0
3.8042  1.2361
4.0420  1.3133
4.7552  1.5451
4.4550  2.2700                                     subregion #2
4.0451  2.9389
3.4383  2.4981
3.2361  2.3511
3.5640  1.8160

        3        4
     2  3     0  0     0  0     0  0
3.2361  2.3511
3.4383  2.4981
4.0451  2.9389
3.5355  3.5355                                     subregion #3
2.9389  4.0451
2.4981  3.4383
2.3511  3.2361
2.8384  2.8284
```



JOIN ARRAY

LOOP#	(1, 1)	(1, 2)	(2, 1)	(2, 2)	(3, 1)	(3, 2)	(4, 1)	(4, 2)
1	0	0	0	0	0	0	0	0
2	1	3	0	0	0	0	0	0
3	2	3	0	0	0	0	0	0
4	3	3	0	0	0	0	0	0
5	4	3	0	0	0	0	0	0

Figure 2.2. Mesh Generation for a Viscoplastic Tube

```
        3      4
     3  3    0  0    0  0    0  0
2.3511   3.2361
2.4981   3.4838
2.9389   4.0451
2.2700   4.4550
1.5451   4.7553
1.3133   4.0420
1.2361   3.8042
1.8160   3.5640
```

subregion #4

```
        3      3
     4  3    0  0    0  0    0  0
1.2361   3.8042
1.3133   4.0420
1.5451   4.7553
.7822    4.9384
0.       5.
0.       4.25
0.       4.
.6257    3.9508
```

subregion #5

The obvious thing to notice in Fig. 2.2 is the higher density of elements obtained along the inside radius of the quarter annulus obtained by placing midside coordinates of the subregions closer to the inside radius. In addition it should be clear that the entire region of study need not have been broken up into so many subregions, but this was done to demonstrate to the user the method of connecting these subregions.

Proceeding, the frontal solution is accomplished by inputting the following data deck to FRONT9:

```
  9                                  read from TAPE9 nodal point coordinates
                                        and element definitions
108  259  259    0    6    6    0    temperature mesh construction

      0                              number of elements in temperature mesh
                                        not composed of material type 1
     -1                              manual element selection for temperature
                                        mesh
  1  108    1                        element ordering in temperature solution

108  442  259  183    9    6    3    viscous mesh construction

      0                              number of elements in viscous mesh not
                                        composed of material type 1
     -1                              manual element selection for viscous mesh

  1  108    1                        element ordering in viscous solution
```

This permits the user to specify the order in which the relevant equations are formulated and solved, and reduces the amount of in-core storage necessary. The actual thermomechanical analyses are then performed with the following cards used as input to PLAST9:

Card input	Description
1	number of materials
.096 1.850.0 0.0 3.38 0.0 3.38 0.0 .001 43900. 0.0 1.0 0.0 0.0 0.0 0.0 0.0	material type 1 definition
0	number of constraints released
500.0 1.0 250. 1.0 3	T_i, conversion factor, T_∞, h, NVPS
36 1 103 6 3 6 12 6 1 18 18 1 2 24 30 6 1 36 42 6 2 48 54 6 1 60 66 6 2 72 78 6 1 84 90 6 2 96 102 6 2 108 108 1 2	surface convection boundary conditions
0.0	decay constant for heat sources (no decay)
14 1 7 1 3 0.0 0.0 0.0 253 258 1 2 0.0 0.0 0.0 259 259 1 2 0.0 -500.0 0.0	applied forces, imposed velocities and temperatures, specified heat fluxes and nodal point restraints
5 5 5 10	print and punch controls
1 -1 1.0	type of solution performed first, coupling, type of analysis
0 2 0 1 30 5 5 1	solution controls
100.0 0.1 0.1 1.0 1.0	maximum problem time and convergence controls
0	do not write ITP1 array
0 0	restart indicator and number of increments to be read from tape 8

Finally, we wish to plot the results. Inputting the following cards into PLOT9 results in Figs. 2.3 through 2.5.

Card	Description
9	read from TAPE9 nodal point coordinates and element definitions
6 1	number of plots and symmetry flag
5 10 15 20 25 30	increment numbers plotted
-7.0 7.0 -7.0 7.0 .50	plot limits and magnification
79 253 1 7 2 6 1 7 259 7	nodal points used as plotting boundaries
1.0 2.0 .50	vector length, increments between isotherms, cutoff velocity

Here, the option of displaying a symmetric half of the region of analysis is selected. Fig. 2.3, the time history of the mesh deformations, shows the continuous flattening of the tube by the applied force. In Fig. 2.4 the time history of the velocity fields shows that only a section of the tube is moving at any given time. This is not true since the velocity field is continuous over the domain, but appears because of the relatively high cutoff velocity, .5, chosen during plotting. Also, from Fig. 2.4, we see that as time progresses more and more of the material velocities exceed the cutoff velocity. Finally in Fig. 2.5, which is the time history of the temperature distributions showing the temperature isotherms, we see the surface cooling due to convective losses and the heating due to viscous dissipation in the vicinity of where the force was applied.

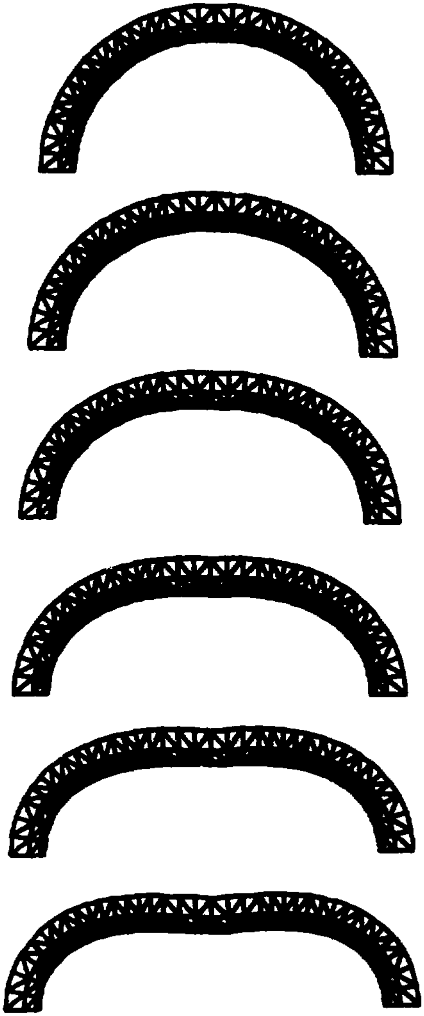


Figure 2.3. Time History of the Mesh Deformation for the Viscoplastic Tube (time progressing from top to bottom)

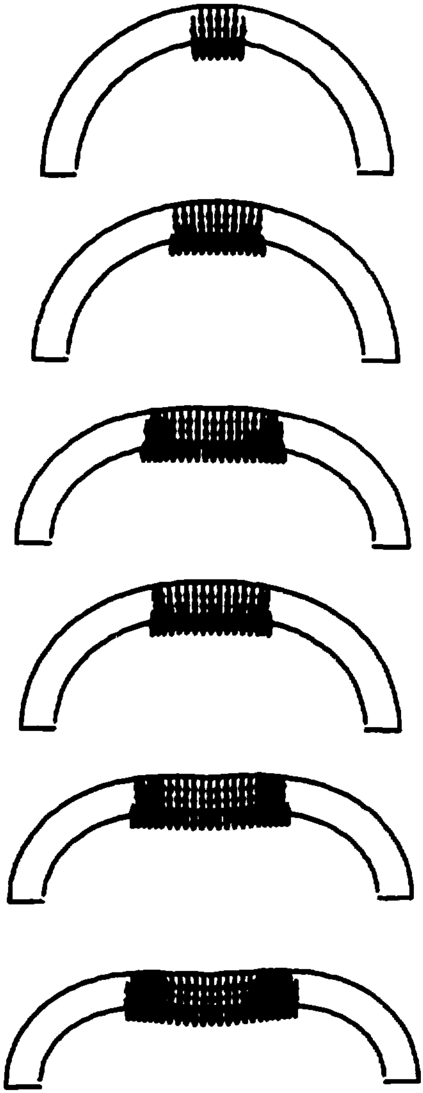


Figure 2.4. Time History of the Velocity Distributions for the Viscoplastic Tube (time progressing from top to bottom)

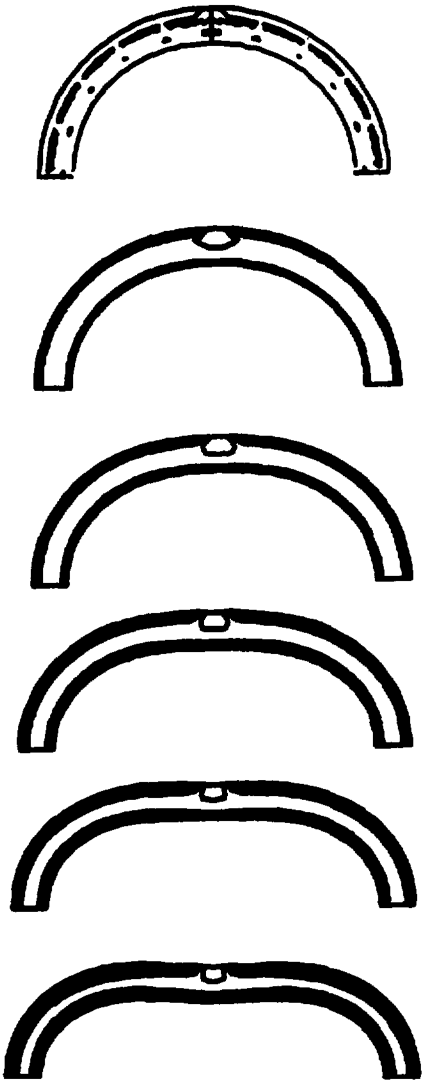


Figure 2.5. Time History of the Temperature Distributions for the Viscoplastic Tube (time progressing from top to bottom)

2.3 Axisymmetric Flow Through a Reducing Duct

The second problem considered was the flow of a soft viscoplastic material through a rigid reducing duct. A cylindrical plug of material is forced through a blunt duct such that the reduced cross section has an area equal to one-third of the plug area. This thermomechanical system we wish to model is depicted in Fig. 2.6. Here, it can be seen that the duct sides are frictionless except along the face perpendicular to the axial direction where no slip occurs. A constant inlet pressure and fixed temperature are imposed on the viscoplastic material entering the duct. The axial thermal gradients on the horizontal surface of both the product and the exposed horizontal surfaces of the duct are zero, while the lateral surfaces of the emerging material and duct are subject to convective heat loss to the surrounding air. Since the container needs to be considered only in the thermal solution, we incorporate the two mesh capability of the code; that is, in the thermal solution we include the entire system, but in the viscous solution only the viscoplastic material itself is included while the rigid duct is neglected.

We begin the solution of this problem as before by dividing the region of analysis into elements. Using the following cards as input to MESH9 yields the plot shown in Fig. 2.7.

	7			number of subregions
0.0	10.0	0.0	18.0	plot limits
	1			slope of diagonal indicator
	9			write to Tape9 nodal point coordinates and element definitions

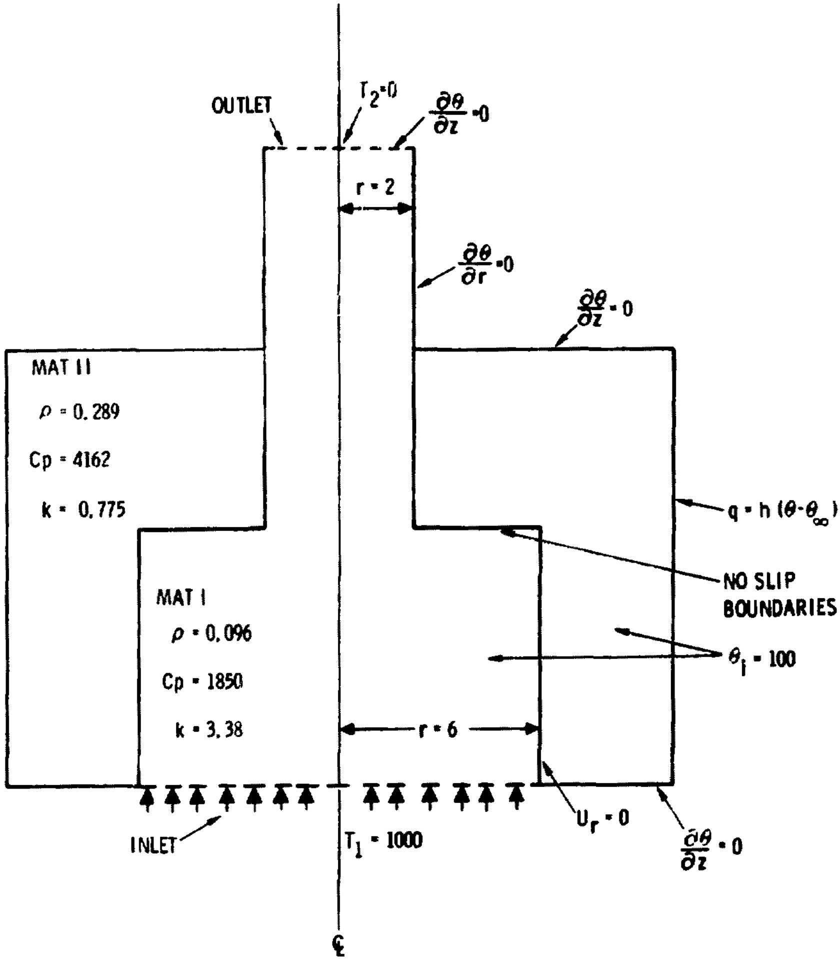
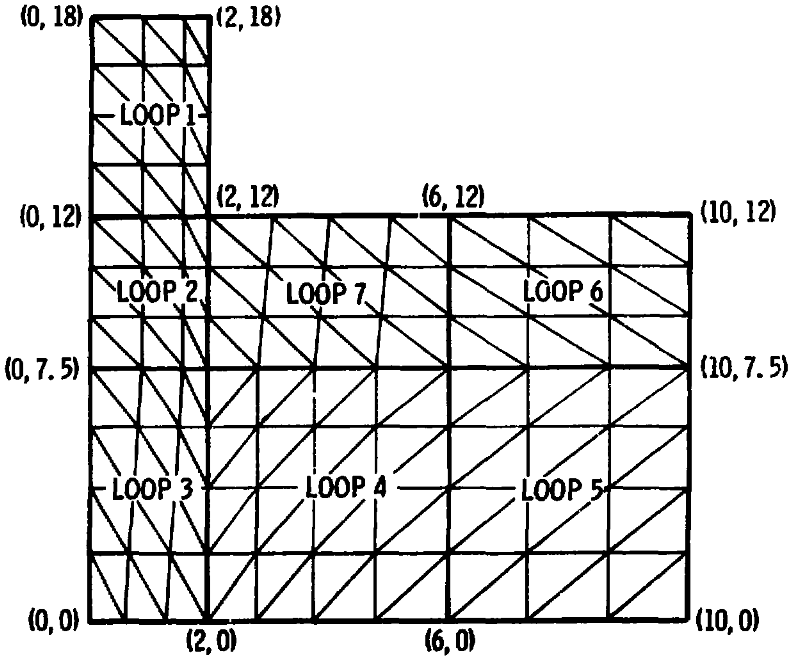


Figure 2.6. Axisymmetric Flow Through a Reducing Duct



JOIN ARRAY

LOOP #	(1, 1)	(1, 2)	(2, 1)	(2, 2)	(3, 1)	(3, 2)	(4, 1)	(4, 2)
1	0	0	0	0	0	0	0	0
2	1	1	0	0	0	0	0	0
3	2	3	0	0	0	0	0	0
4	3	4	0	0	0	0	0	0
5	4	3	0	0	0	0	0	0
6	5	4	0	0	0	0	0	0
7	2	4	4	4	6	4	0	0

Figure 2.7. Mesh Generation for Flow Through a Reducing Duct

```
        2         4
   0    0    0    0    0    0       0    0    } Number of divisions, connection
 0.0       12.0                               } to previous subregions, and sub-
 1.25      12.0                               } region coordinates for:
 2.0       12.0                               }
 2.0       15.0                               } subregion #1
 2.0       18.0                               }
 1.25      18.0                               }
 0.0       18.0                               }
 0.0       15.0

        3         3
   1    1    0    0    0    0       0    0    }
 2.0       12.0                               }
 1.25      12.0                               }
 0.0       12.0                               }
 0.0        9.75                              } subregion #2
 0.0        7.5                               }
 1.25       7.5                               }
 2.0        7.5                               }
 2.0        9.75                              }

        3         4
   2    3    0    0    0    0       0    0    }
 2.0        7.5                               }
 1.25       7.5                               }
 0.0        7.5                               } subregion #3
 0.0        4.1                               }
 0.0        0.0                               }
 1.00       0.0                               }
 2.0        0.0                               }
 2.0        4.1

        4         4
   3    4    0    0    0    0       0    0    }
 2.0        7.5                               }
 2.0        4.1                               }
 2.0        0.0                               } subregion #4
 3.75       0.0                               }
 6.0        0.0                               }
 6.0        3.9                               }
 6.0        7.5                               }
 3.75       7.5

        4         3
   4    3    0    0    0    0       0    0    }
 6.0        7.5                               }
 6.0        3.9                               }
 6.0        0.0                               } subregion #5
 8.0        0.0                               }
10.0        0.0                               }
10.0        3.9                               }
10.0        7.5                               }
 8.0        7.5
```

```
      3       3
  5   4    0  0    0   0       0   0   }
 6.0      7.5                          }
 8.0      7.5                          }
10.0      7.5                          }  subregion #
10.0      9.75                         }
10.0     12.0                          }
 8.0     12.0                          }
 6.0     12.0                          }
 6.0      9.75
      3       4
  2   4    4  4    6   4       0   0   }
 2.0     12.0                          }
 2.0      9.75                         }
 2.0      7.5                          }  subregion #7
 3.75     7.5                          }
 6.0      7.5                          }
 6.0      9.75                         }
 6.0     12.0                          }
 4.0     12.0
```

It is essential to construct the mesh such that the region in which the viscous flow will occur will be the first elements defined. Thus, when the number of elements are specified in the frontal solution program for both the temperature and viscous analyses the correct elements will be obtained. This may be made clearer by referencing Fig. 2.8 and 2.9 which are obtained from PLOT9. Fig. 2.8 shows the element mesh for the viscous solution and its symmetric half about the centerline. This mesh is composed of the first NUMEL2 elements generated by MESH9 where NUMEL2 is explained in Chapter 4. Fig. 2.9 shows the element mesh for the temperature solution, and its symmetric half about the centerline. This mesh is composed of NUMEL1 elements where again NUMEL1 is explained in Chapter 4.

Proceeding as before the frontal solutions are accomplished with the following data cards:

```
  9                                read from TAPE9 nodal point coordinates and
                                      element definitions
164  371  371   0  6  6  0         temperature mesh construction
```

66	number of elements not composed of material type 1 in thermal mesh
99 164 1 2	remaining element numbers and material numbers
0	automatic element selection for temperature mesh
24 1	in automatic element selection start with element number 24 and pick the following elements such that the most nodal points are completed
98 536 371 165 9 6	viscous mesh construction
0	number of elements not composed of material type 1 in viscous mesh
0	automatic element selection for viscous mesh
24 1	in automatic element selection start with element number 24 and pick the following elements such that the most nodal points are completed

Here, again, the importance of the mesh construction is emphasized. In the first frontal solution accomplished, that associated with the temperature analysis, all 164 elements are used. In the second frontal solution, that associated with the viscous flow problem, only the first 98 elements are used. Thus the 2 mesh capability of COUPLEFLO is exercised.

The thermomechanical solution is accomplished by inputting the following cards into PLASTO.

2								number of materials
.096	1850.0	0.0	3.38	0.0	3.38	0.0	.1E-11	material type 1 definition
97500.00	0.0	.20	0.0	0.0	0.0	0.0	0.0	
.28[illegible]	4162.0	0.0	.775	0.0	.775	0.0	0.0	material type 2 definition
0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
0								number of constraints released
100.	1.0	0.0	0.0	3				T_i, conversion factor, T_∞, h, NVPS
7								surface convection boundary conditions
116 122	2	2						
128 140	6	1						
0.0								half life for heat sources (no decay)

	59						
161	161	1	-1	0.0	-.111334E2	100.0	applied forces, imposed velocities and temperatures, specified heat fluxes and nodal point restraints
160	160	1	-1	0.0	.1482967E5	100.0	
159	159	1	-1	0.0	.1482962E5	100.0	
158	158	1	-1	0.0	.4440000E5	100.0	
157	157	1	-1	0.0	.2967028E5	100.0	
156	156	1	-1	0.0	.7410387E5	100.0	
155	155	1	-1	0.0	.4669241E5	100.0	
170	170	1	-1	0.0	.1306222E6	100.0	
179	179	1	-1	0.0	.8093431E5	100.0	
188	188	1	-1	0.0	.2052090E6	100.0	
197	197	1	-1	0.0	.1237252E6	100.0	
206	206	1	-1	0.0	.3036739E6	100.0	
215	215	1	-1	0.0	.1790498E6	100.0	
224	224	1	-1	0.0	.4281264E6	100.0	
233	233	1	-2	0.0	.1241445E6	100.0	
226	232	1	2	0.0	0.0	0.0	
162	225	9	4	0.0	0.0	0.0	
64	99	7	4	0.0	0.0	0.0	
7	7	1	4	0.0	0.0	0.0	
1	57	7	2	0.0	0.0	0.0	
70	160	7	2	0.0	0.0	0.0	

1	1							print and punch controls
1	1							
1	-1	0.0						type of solution performed first, coupling, type of analysis
1	-2	1	1	1	2	3	0	solution controls
100.	.001	.1	.001	.001				maximum problem time and convergence controls
0								do not write ITP1 array
0	0							restart indicator and number of increments to be read

Here the assumed constitutive equation for the flowing viscoplastic material is

$$\sigma'_{II} = 97500\, \epsilon_{II}^{1/5} \qquad (2.2)$$

while none is needed for the duct and container since they are not included in the viscous solution. In addition, the material properties for the flowing viscoplastic material are taken as those used in the compression of the viscoplastic tube. The container is assumed to have a density of .289, a heat capacity of 4162, and a constant conductivity of .775. The description of the reduction system is completed by the specification of the no slip conditions along the interior of the die. Finally only the steady state solution is required.

After completion of the thermomechanical solution of the problem, plots may be obtained by inputting the following cards into PLOT9.

9	read nodal point coordinates and element definitions from TAPE9
1 1	number of plots and symmetry
2	increment numbers plotted
-15.0 10.0 0.0 20.0 1.	plot limits and magnification
94 57 63 1 56 7 7 330 366 6 323 329 1 322 294 7 279 287 1 278 170 9 155 161 1 160 155 1 170 233 9 232 225 1 216 162 9 99 99 1 7 7 1	nodal points used for plot boundaries
.5 5. .000001	vector length, increments between isotherms, cutoff velocity

Again the option to plot the symmetric half of the region of analyses is selected. From these input cards we obtain Figs. 2.8 through 2.11 which show the element meshes used for the reduction system, along with the steady state velocity field and temperature distributions obtained. As previously mentioned, Figs. 2.8 and 2.9 show the meshes used for the viscous and temperature analysis. Fig. 2.10 is the steady state velocity profile which reveals the stationary layer of material in the vicinity of where the no slip conditions were imposed. Also from Fig. 2.10 we see the parabalic nature of the magnitude of the velocities, which effects the temperature distributions, both from convection and viscous dissipation. Fig. 2.11 shows the steady state temperature distribution of the isotherms. The effect of the parabolic velocity distribution becomes evident as isotherms are carried ahead along the centerline. Also, the high thermal gradients near the inlet of the duct are seen together with the surface cooling of the duct from the convective losses.

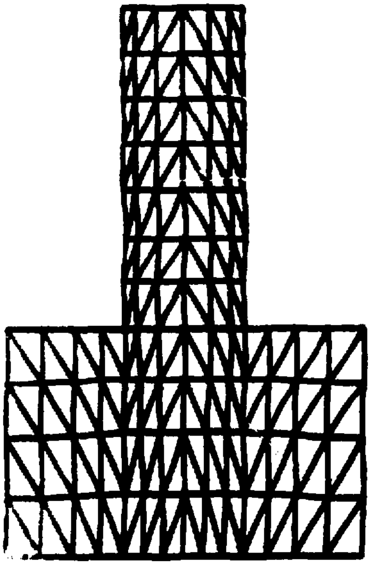


Figure 2.8. Viscous Mesh for Flow Through a Reducing Duct

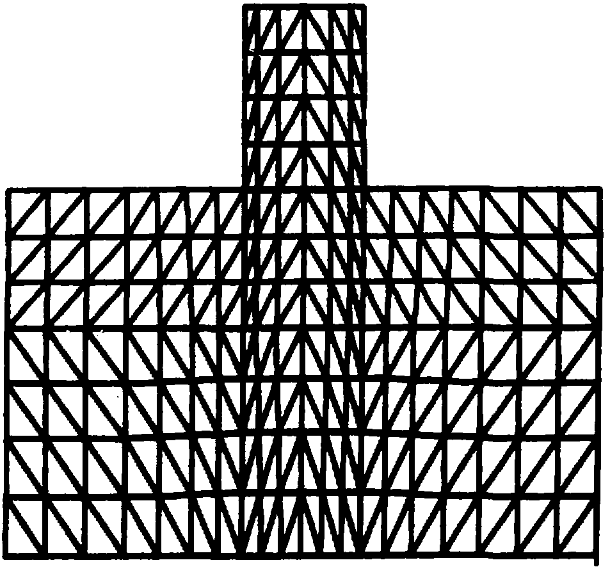


Figure 2.9. Temperature Mesh for Flow Through a Reducing Duct

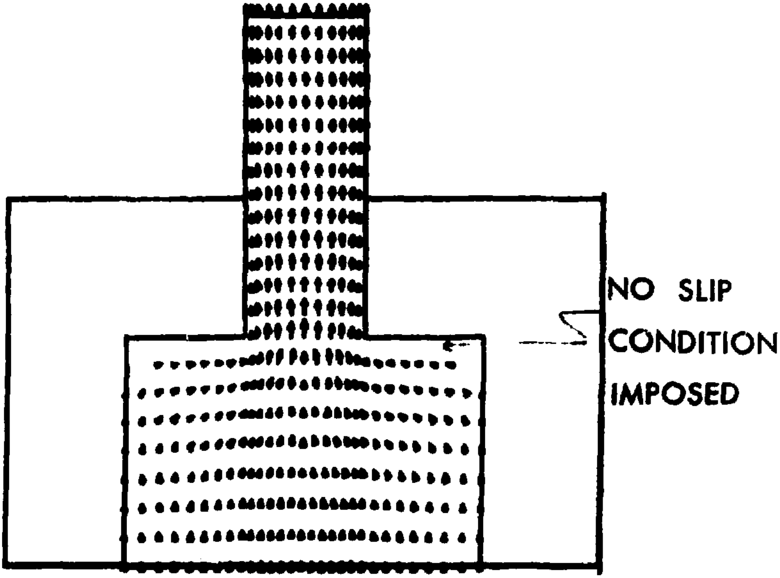


Figure 2.10. Velocity Field for Flow Through a Reducing Duct

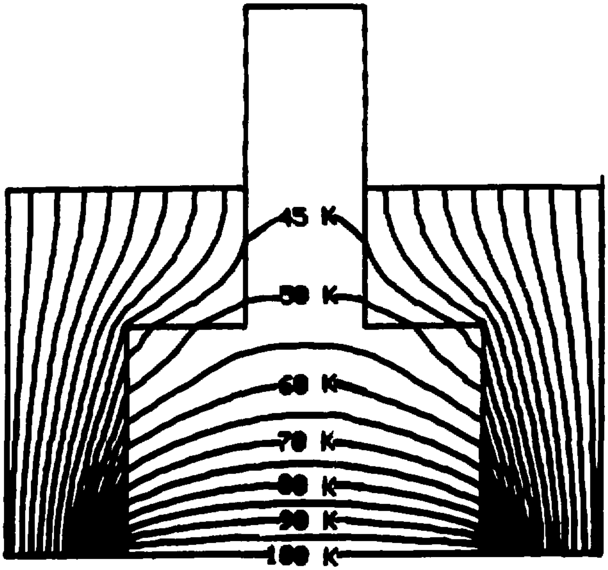


Figure 2.11. Temperature Distribution for Flow Through a Reducing Duct

Chapter III

EXECUTION INSTRUCTIONS FOR COUPLEFLO

The current COUPLEFLO program has been subdivided into four separate programs named MESH9, FRONT9, PLAST9, and PLOT9. The function of those four programs is to respectively generate the mesh, set up the frontal solution, solve the thermomechanical problem and plot the results. The primary reasons behind the substructuring of the COUPLEFLO code was to 1) allow the user to execute the program in stages which allow for verification of separate steps, 2) allow the user convenient entry points into the code to permit efficient parameter studies, 3) and to reduce the central memory requirement which improves turnaround time.

The substructuring of the COUPLEFLO program necessitates a communication between each of the subprograms. As will be seen in the appendix containing the input instruction, a few options are available to the user as to how information will be passed from one program to the next. In other cases, no option exists for the transfer of data, and the user must either regenerate the necessary data for subsequent problem execution or, after initial data generation, may store it in permanent file space. An example of the option in transferring data is the nodal point coordinates and element definitions calculated in MESH9. This information is necessary in both the frontal solution and the plotting programs. An option exists as to whether this information is written on to a disk file, TAPE9, which is suitable for cataloging, or whether the user will obtain a punched deck of the data. An example where no option is available is the frontal solution arrays for the temperature and viscous flow solutions, which are written onto TAPE1 and TAPE2, respectively. Presently the only way of saving these files, which are necessary in PLAST9, is to catalog them onto permanent file space.

The division of the COUPLEFLO code also makes for a longer control card sequence. The following control stream represents a possible execution of the entire COUPLEFLO program on the 6600 scope system.

```
 1.  JOB CARD.
 2.  ACCOUNT CARD.
 3.  REQUEST,TAPE7,*PF.
 4.  REQUEST,TAPE9,*PF.
 5.  ATTACH,A,MESH9.                          }
 6.  UPDATE,P=A,N,F,C=CA,W.                   } Load and execute the
 7.  FTN,I=CA,L=0,B=BA.                       } mesh generator
 8.  COLLECT,LGO,DISSPLA,SMALL42,FTNLIB.      }
 9.  BA.                                      }
10.  CATALOG,TAPE9,ANY FILE NAME,CN=ANY PASSWORD. } Save nodal point
                                                    coordinates and
                                                    element definitions
11.  REWIND,TAPE9.
12.  RFL,135000.
13.  REQUEST,TAPE1,*PF.
14.  REQUEST,TAPE2,*PF.
15.  REQUEST,TAPE10,*PF.
16.  ATTACH,B,FRONT9.          }
17.  UPDATE,P=B,N,F,C=CB,W.    } Load and execute frontal solution
18.  FTN,I=CB,L=0,B=BB.        } program
19.  COLLECT,LGO,FTNLIB.       }
20.  BB.                       }
21.  CATALOG,TAPE1,ANY FILE NAME,CN=ANY PASSWORD.  } Save ITP1 arrays
22.  CATALOG,TAPE2,ANY FILE NAME,CN=ANY PASSWORD.  } for thermal and
23.  CATALOG,TAPE10,ANY FILE NAME,CN=ANY PASSWORD. } viscous meshes
                                                     along with the
                                                     necessary common
                                                     block
```

```
24.  REWIND,TAPE1.
25.  REWIND,TAPE2.
26.  REWIND,TAPE10.
27.  RFL,135000.
28.  ATTACH,C,PLAST9                 }
29.  UPDATE,P=C,N,F,C=CC,W.          }
30.  FTN,I=CC,L=0,B=BC.              }  Load and execute the
                                     }  thermomechanical solver
31.  COLLECT,LGO,FTNLIB.             }
32.  BC.                             }
33.  RETURN,TAPE1.
34.  RETURN,TAPE2.
35.  RETURN,TAPE10.
36.  REWIND,TAPE9.
37.  RFL,135000.
38.  ATTACH,D,PLOT9.                                }
39.  UPDATE,P=D,N,F,C=CD,W.                         }
40.  FTN,I=CD,L=0,B=BD.                             }  Load and execute
                                                    }  the plotting program
41.  COLLECT,LGO,DISSPLA,SMALL42,FTNLIB.            }
42.  BD.                                            }
43.  END OF RECORD CARD.
44.  UPDATE DECK FOR MESH9.
45.  END OF RECORD CARD.
46.  DATA DECK FOR MESH9.
47.  END OF RECORD CARD.
48.  UPDATE DECK FOR FRONT9.
49.  END OF RECORD CARD.
50.  DATA DECK FOR FRONT9.
```

51. END OF RECORD CARD

52. UPDATE DECK FOR PLAST9.

53. END OF RECORD CARD.

54. DATA DECK FOR PLAST9.

55. END OF RECORD CARD.

56. UPDATE DECK FOR PLOT9.

57. END OF RECORD CARD

58. DATA DECK FOR PLOT9

59. END OF INFORMATION CARD

60. END OF INFORMATION CARD

Here we are assuming that the nodal point coordinates have been written onto TAPE9 and we wish to save them. Furthermore, the contents of TAPE1 and TAPE2, as noted previously are to be cataloged. TAPE10 which contains a common block defined in FRONT9 and is needed in PLAST9 is also saved for later use. TAPE7 which is written onto by MESH9 and PLOT9 must also be equivalenced to a permanent file in order to obtain hard copy plots. Obviously the user may execute any of the four separate programs by accessing the necessary data, if any, and executing that specific program, which is the recommended procedure. Similarly, the following control stream represents a possible execution of the entire COUPLEFLO program on the 7600 scope system with the identical options used in the 6600 control stream:

1. JOB CARD.

2. ACCOUNT CARD.

3. ATTACH,DISSPLA,DISSPLA.

4. ATTACH,FXSCORS,FXSCORS.

5. LIBRARY,DISSPLA,FXSCORS.

6.	FILE,A,RT=S.	Load and execute the mesh generator
7.	ATTACH,A,MESH9.	
8.	UPDATE,P=A,F,W,C=CA.	
9.	FTN,I=CA,B=BA.	
10.	BA.	
11.	CATALOG,TAPE9,ANYFILE NAME,CN=ANY PASSWORD.	Save nodal point coordinates and element definitions
12.	FILE,B,RT=S.	
13.	ATTACH,B,FRONT9.	Load and execute frontal solution program
14.	UPDATE,P=B,F,W,C=CB.	
15.	REWIND,TAPE9.	
16.	FTN,I=CB,B=BB.	
17.	BB.	
18.	CATALOG,TAPE1,ANY FILE NAME,CN=ANY PASSWORD.	Save ITP1 arrays for thermal and viscous meshes along with the necessary common block
19.	CATALOG,TAPE2,ANY FILE NAME,CN=ANY PASSWORD.	
20.	CATALOG,TAPE10,ANY FILE NAME,CN=ANY PASSWORD.	
21.	FILE,C,RT=S.	Load and execute the thermo-mechanical solver
22.	ATTACH,C,PLAST9.	
23.	UPDATE,P=C,F,W,C=CC.	
24.	REWIND,TAPE1.	
25.	REWIND,TAPE2.	
26.	REWIND,TAPE10.	
27.	FTN,I=CC,B=BC.	
28.	BC.	
29.	FILE,D,RT=S.	
30.	ATTACH,D,PLOT9.	Load and execute the plotting program
31.	UPDATE,P=D,F.W,C=CD.	
32.	REWIND,TAPE9.	

33. FTN,I=CD,B=BD.
34. BD.

} Load and execute the plotting program

35. END OF RECORD CARD.

36. UPDATE DECK FOR MESH9.

37. END OF RECORD CARD.

38. DATA DECK FOR MESH9.

39. END OF RECORD CARD.

40. UPDATE DECK FOR FRONT9.

41. END OF RECORD CARD.

42. DATA DECK FOR FRONT9.

43. END OF RECORD CARD.

44. UPDATE DECK FOR PLAST9.

45. END OF RECORD CARD.

46. DATA DECK FOR PLAST9.

47. END OF RECORD CARD.

48. UPDATE DECK FOR PLOT9.

49. END OF RECORD CARD.

50. DATA DECK FOR PLOT9.

51. END OF INFORMATION CARD.

52. END OF INFORMATION CARD.

COUPLEFLO is also available on the Sandia 6600 NOS Time Sharing Computer. A procedure file has been written to allow the user to execute one of the subprograms comprising COUPLEFLO or a sequence of the subprograms. Also, options exist as to whether the users input files and a few of his output files used to execute the subsequent programs are saved on his permanent file space. Finally, it is also possible for the user to receive a printer copy of the output from each of the executed programs. To select the desired options, the user must set three internal registers on the NOS system. These three registers have the following functions:

R1 IS A FLAG USED TO DETERMINE WHICH PROGRAMS WILL BE COMPILED AND/OR RUN.

IF R1=1 THEN THE PLOTTING PROGRAM IS COMPILED AND EXECUTED

IF R1=2 THEN IT IS ASSUMED A COMPILED VERSION OF THE PLOTTING PROGRAM EXISTS AND IT IS EXECUTED

IF R1=4 THEN THE VISCOUS/TEMPERATURE SOLUTION PROGRAM IS COMPILED AND EXECUTED FOLLOWED BY THE RESULTS OF SETTING R1=1

IF R1=5 THEN THE RESULTS ARE THE SAME AS R1=4 EXCEPT IT IS ASSUMED THAT A COMPILED VERSION OF THE VISCOUS/TEMPERATURE SOLUTION EXISTS

IF R1=6 THEN THE PROGRAM CALCULATING THE FRONTAL SOLUTION WILL BE COMPILED AND EXECUTED FOLLOWED BY THE RESULTS OF SETTING R1=4

IF R1=7 THEN THE RESULTS ARE THE SAME AS R1=6 EXCEPT IT IS ASSUMED THAT A COMPILED VERSION OF THE FRONTAL SOLUTION PROGRAM EXISTS

IF R1=8 THEN THE PROGRAM CALCULATING THE MESH WILL BE COMPILED AND EXECUTED FOLLOWED BY THE RESULTS OF SETTING R1=6.

IF R1=9 THEN THE RESULTS ARE THE SAME AS R1=8 EXCEPT IT IS ASSUMED THAT A COMPILED VERSION OF THE MESH CALCULATION PROGRAM EXISTS

FURTHERMORE BY SETTING R1 TO THE NEGATIVE OF ANY OF THE ABOVE VALUES PROGRAM EXECUTION TERMINATES WITH THAT PARTICULAR PROGRAM. THIS ALLOWS FOR PROGRAM AND/OR OUTPUT VERIFICATION FOR ANY PARTICULAR PROGRAM. THE USER IS ADVISED TO USE THIS OPTION UNLESS HE HAS SOME PREVIOUS INDICATION THE PROGRAM WILL WORK IN ITS ENTIRETY.

REGISTER R2 IS USED TO DETERMINE IF THE USER WISHES A PRINTOUT OF ANY OF THE RESULTS OBTAINED FROM ALL OF THE PROGRAMS EXECUTED. THE OUTPUT IS OBTAINED IN THE TERMINAL ROOM IN BUILDING 806--ROOM 159.

IF R2=0 THEN NO PRINTOUT IS OBTAINED

IF R2=1 THEN A PRINTOUT IS OBTAINED

REGISTER R3 IS USED TO DETERMINE IF THE USER WISHES TO SAVE IN HIS PERMANENT FILE SPACE ALL INPUT DATA TO THE PROGRAMS AND INFORMATION GENERATED BY AN EXECUTED PROGRAM THAT IS USED IN EXECUTING ANY SUBSEQUENT PROGRAM.

IF R3=0 NOTHING WILL BE STORED IN THE USERS PERMANENT FILE SPACE

IF R3=1 THEN ALL INPUT DATA AND INFORMATION USED IN EXECUTING SUBSEQUENT PROGRAMS WILL BE SAVED IN THE USERS PERMANENT FILE SPACE

FINALLY IN ORDER TO SET ANY OF THESE OPTIONS THE USER MUST TYPE THE FOLLOWING; SET(RI=N). THIS ASSIGNS TO REGISTER I THE VALUE OF N

EXAMPLE: SUPPOSE THE USER WISES TO ONLY EXECUTE THE MESH GENERATION PROGRAM AND DOES NOT WANT A PRINTOUT OF THE OUTPUT BUT DOES WANT TO SAVE ALL FILES NECESSARY IN RERUNNING THE MESH PROGRAM AND EXECUTING THE SUBSEQUENT PROGRAMS. THE USER THEN SETS THE THREE REGISTERS IN LIKE MANNER

SET(R1=-8)
SET(R2=1)
SET(R3=0)

THE NOS SYSTEM WILL ECHO EACH COMMAND BACK TO THE USER.

Due to the limits on permanent file space on NOS at this time it is impossible to save the frontal solution arrays and the common block defined in FRONT9. Therefore, the user must rerun this program every time he wishes to execute PLAST9, or have them saved on a mountable magnetic tape.

Finally to invoke the procedure file the user must specify the names of the files that contain all necessary input to any of the programs executed. This is done at the time of execution of the procedure file by equivalencing a code letter to the file name containing the necessary input. The code letter and the file associated with that code letter are described below. Hence, if the user wishes to only execute the mesh generation program and obtain a printed output, without saving the file containing the nodal point coordinates and element definitions he would set the necessary registers, as shown in the above example, and call the procedure file by typing

-UCOUPLE(L=FN)

where FN is the users file name containing the input to MESH GENERATOR.

L = File Name containing the input to the mesh generation program

M = File name containing the nodal point coordinates and element definitions if the user selects the option to save this data in permanent file space, (SET(R3=1))

N = File name containing the input to the frontal program

O = File name containing the input to the viscous and temperature solution program

P = File name containing the plot data if the user selects the option to save this data in permanent file space

Q = File name containing the input to the plotting program

Program Executed	L	M	N	O	P	Q
MESH9	*	✓				
FRONT9		*	*			
PLAST9				*	✓	
PLOT9		*			*	*

(Column heading over L–Q: Files Required)

Asterisks represent necessary files to the program executed.

Checks represent files necessary to the program executed with the R3 register set to one option.

Chapter IV

INPUT INSTRUCTIONS FOR COUPLEFLO

4.1 Introduction

The next few pages to be found in this chapter are instructions to data input into each of the four separate programs comprising COUPLEFLO. The following conventions apply for each card:

1) Following the card number and the format in which the data is typed a brief description of the information contained on the card is given. Included in this information are the circumstances as to whether or not this card is contained in the input deck and how many times it is read.

2) The subscripts on a variable name indicate the columns in which it is to be punched, e.g., ${}_{11}\mathrm{Var}_{20}$ indicates Var is to be punched in columns 11 through 20, according to the specified format.

3) If a coded response is necessary in any of the input, the options allowable and the results of setting the option to a particular value are displayed directly after the description of the variable being considered.

4) Very few default values are incorporated into any four subprograms that comprise COUPLEFLO and the user is strongly urged to specify all variables explicitly. Any variable that is defaulted though is shown in the input instruction as the default value in parenthesis following the variable description.

5) Card inputs to each of four subprograms must be sequential; that is, Card 2 must follow Card 1 and if Card 7 is omitted Card 8 must follow Card 6.

6) In addition, when the user selects coupling between the mechanical and thermal systems, he is advised to begin first with the solution of the mechanical system. This is always the case unless there are no initially prescribed forces or nonzero velocities whereupon the user may then begin with the solution to the thermal system.

4.2 Input Instructions for MESH9

Card 1 Format (I10)

Card 1 specifies the number of subregions into which the entire domain being studied will be divided. This is particularly useful in obtaining better resolution in particular areas by requiring more elements in that area.

Var Name - Description

$_{1}$NUMLPS$_{10}$ - the number of subregions in the domain being studied (maximum of 14)

Card 2 Format (4F10.3)

Card 2 provides dimensions for the plotted data. The bounds of the region may correspond to the size of the entire region being studied or may be some subset of that region.

Var Name - Description

$_{1}$XMIN$_{10}$ - minimum X coordinate to be plotted

$_{11}$XMAX$_{20}$ - maximum X coordinate to be plotted

$_{21}$YMIN$_{30}$ - minimum y coordinate to be plotted

$_{31}$YMAX$_{40}$ - maximum y coordinate to be plotted

Card 3 Format (I10)

MESH9 will normally draw the diagonal through the rectangle it constructs, for element formulation, such that the length of the diagonal is a minimum. In the case where the length of the diagonals are the same Card 3 allows the user to specify which way the diagonal will be drawn.

Var Name - Description

$_1$IDIAG$_{10}$ - user option for specifying how diagonal is drawn

1 - with negative slope

2 - with positive slope

Card 4 Format (I10)

Card 4 allows the user to either obtain a punched deck or a disk file of the MESH9 data which is necessary input for FRONT9 and PLOT9.

Var Name - Description

$_1$IOUTPUT$_{10}$ - option to write MESH9 generated data to a punch or disk file

8 - output goes to punch file

9 - output goes to disk file which is suitable to be cataloged in permanent file space

Cards 5 through 7 are read as a set NUMLPS times and contain information on each subregion of the area of analysis.

Card 5 Format (2I10)

Card 5 allows the user to specify the number of rectangles in a particular subregion. The user can then determine how many triangular elements will be drawn in the subregion by multiplying the number of rectangles by 2.

Var Name - Description

$_1$NDIV(I,1)$_{10}$ - the number of divisions on sides 1 and 3 for the Ith subregion (maximum of 13)

$_{11}$NDIV(I,2)$_{20}$ - the number of divisions on sides 2 and 4 for the Ith subregion (maximum of 13)

Card 6   Format (4(I7,I3))

Card 6 specifies the manner in which the subregions will be connected. Since Card 6 is read NUMLPS times the information of the I^{th} Card 6 will pertain to how the I^{th} subregion will be connected to subregions already generated. It is necessary to specify data for connecting the I^{th} subregion to the previous I-1 subregions but the user should not specify how the I^{th} subregion will be connected to subsequent subregions to be generated.

<u>Var Name</u> - <u>Description</u>

${}_{1}JOIN(I,1,1)_{7}$ - which subregion the first side of the I^{th} subregion will be connected to

${}_{8}JOIN(I,1,2)_{10}$ - which side the first side of the I^{th} subregion will be connected to

${}_{11}JOIN(I,2,1)_{17}$ - which subregion the second side of the I^{th} subregion will be connected to

${}_{18}JOIN(I,2,2)_{20}$ - which side the second side of the I^{th} subregion will be connected to

${}_{21}JOIN(I,3,1)_{27}$ - which subregion the third side of the I^{th} subregion will be connected to

${}_{28}JOIN(I,3,2)_{30}$ - which subregion the third side of the I^{th} subregion will be connected to

${}_{31}JOIN(I,4,1)_{37}$ - which side the fourth side of the I^{th} subregion will be connected to

${}_{38}JOIN(I,4,2)_{40}$ - which side the fourth side of the I^{th} subregion will be connected to

Card 7   Format (2F10.3)

Card 7 is read 8 times for each subregion. It contains one of the coordinates that defines the subregion currently being generated. These

8 coordinates must be entered in counter-clockwise fashion from a starting point at one corner of the subregion chosen by the user, which also defines the sides of the region. Four of the eight coordinates must be the locations of the corners of the subregion and the remaining 4 coordinates are locations of points between the corners. If the user chooses the midside coordinates to be midway between the corners of each side, a uniformly spaced mesh of triangular elements is obtained. If the midside coordinates are chosen to be closer to one corner than the other, a higher density of elements is obtained in a portion of the subregion. Again the need to enter the 8 coordinates in a counter-clockwise direction beginning with a corner is emphasized.

Var Name - Description

$_{1}$XCOR(I,J)$_{10}$ - the Jth x-coordinate of the Ith region

$_{11}$YCOR(I,J)$_{20}$ - the Jth y-coordinate of the Ith region

NOTE: COUPLEFLO is currently capable of handling up to 300 elements which may consist of up to 700 velocity points and 500 pressure points. Increased dimensions of several arrays will be needed if these parameters are exceeded.

4.3 Input Instructions for FRONT9

Card 1   Format (1I5)

Card 1 allows the user the option to either read in the generated data from MESH9 as cards in his data deck or from a disk file.

Var Name - Description

$_{1}$NTAPE$_{5}$ - the unit number from which MESH9 data will be read

5 - cards in the data deck

9 - Disc File TAPE9

Card 2   Format (7I5)

Card 2 provides information for construction of the first mesh, which is used in the temperature solution.

Var Name - Description

$_{1}$NUMEL1$_{5}$ - Number of elements in the first mesh (maximum of 300)

$_{6}$NUMNP1$_{10}$ - Number of nodal points in the first mesh

$_{11}$NUMVP1$_{15}$ - Number of velocity points in the first mesh (maximum of 700)

$_{16}$NUMPP1$_{20}$ - Number of pressure points in the first mesh (0)

$_{21}$NNPE1$_{25}$ - Number of nodal points per element in the first mesh (6)

$_{26}$NVPE1$_{30}$ - Number of velocity points per element in the first mesh (6)

$_{31}$NPPE1$_{35}$ - Number of pressure points per element in the first mesh (0)

Card 3   Format (I10,2F10.3)

Card 3 is read NUMVP1 times and contains the nodal point coordinates (usually generated by MESH9). If the user has selected the option to read this data from disk, these cards are not contained in the card deck.

Var Name - Description

$_{1}I_{10}$ - Nodal point number

$_{11}XORD(I)_{20}$ - X-coordinate of nodal point I

$_{21}YORD(I)_{30}$ - Y-coordinate of nodal point I

Card 4   Format (10I7)

Card 4 is read NUMEL1 times and contains the nodal point numbers and the pressure point numbers that comprise each of the elements (again this data is usually generated by MESH9). If the user has selected the option to read this data from disk, these cards are not contained in the card deck.

Var Name - Description

$_{1}J1_{7}$ - Element number

$_{8}NPA(J1,1)_{14}$ - First nodal point number comprising the J1 element

$_{8}NPA(J1,2)_{21}$ - Second nodal point number comprising the J1 element

. .

. .

. .

$_{43}NPA(J1,6)_{49}$ - Sixth nodal point number comprising the J1 element

$_{50}NPA(J1,7)_{56}$ - First pressure point number comprising the J1 element

$_{57}NPA(J1,8)_{63}$ - Second pressure point number comprising the J1 element

$_{64}NPA(J1,9)_{70}$ - Third pressure point number comprising the J1 element

Card 5   Format (I10)

The material reference number array, MATA, is initialized to all 1's for the temperature mesh by the program. Card 5 allows the user to change the material composition of any of the elements he desires.

Var Name - Description

${}_{1}\text{NOTONE}_{10}$ - Number of elements in the temperature mesh that are not composed of material reference number 1

Card 6   Format (4I5)

If NOTONE (Card 5) has previously been set to 0, then card 6 must not appear in the card deck. Otherwise card 6 contains information on the material composition of the elements in the temperature mesh. Card 6 is read as many times as necessary until the material composition of all NOTONE elements are specified.

Var Name - Description

${}_{1}\text{I}_{5}$ - the first element number in a set to be changed from material reference number 1

${}_{6}\text{J}_{10}$ - the last element number in a set to be changed from material reference number 1

${}_{11}\text{K}_{15}$ - the increment that will be used between elements I and J in changing the material reference number (the same reference number)

${}_{16}\text{L}_{20}$ - the new material reference number for elements I through J by increments of K

Card 7   Format (I10)

Card 7 offers the choice of element selection options for the frontal solution method in the temperature method. The user may specify either that the program choose element order automatically or that manual (user) ordering will be provided.

Var Name - Description

${}_{1}\text{ISTYLE}_{10}$ - method of element selection

0 - automatic element selection

1 or -1 - manual (user) element selection

Card 8 Format (2I10)

If ISTYLE (Card 7) has been set to 1 or -1, manual element selection, then card 8 must not appear in the card deck. Otherwise card 8 specifies which element should be taken first in the frontal solution. The user may then specify whether the selection of succeeding elements be based on adding elements that will complete the highest number of nodal points or adding the element that will add the fewest number of new nodal points to the coefficient matrix.

Var Name - Description

$_{1}IELE_{10}$ - First element to be taken in the frontal solution method

$_{11}ISEL_{20}$ - Element selection option

1 - selects element that adds least number of new nodal points to the coefficient (stiffness) matrix

2 - selects element that completes most nodal points

Card 9 Format (3I5 for ISTYLE = -1; 20I4 for ISTYLE = 1)

If ISTYLE (Card 7) has been set to 0, automatic element selection, then card 9 must not appear in the card deck. Otherwise card 9 contains the element numbers in the order that they are to be added to the coefficient matrix for the temperature mesh. If ISTYLE = -1 card 9 is read for sequential sets of element numbers as many times as necessary until the order of NUMEL1 elements are specified. If ISTYLE = 1 then all the element numbers are specified individually.

Var Name - Description (for ISTYLE = -1)

$_{1}I_{5}$ - the first element in a set added to coefficient matrix in the frontal solution

$_{6}J_{10}$ - the last element in a set added to coefficient matrix in the frontal solution

$_{11}K_{15}$ - the magnitude of the increment that will be used between elements I and J in adding the new elements to the coefficient matrix in the frontal solution (numbering may be done forwards or backwards)

Var Name - Description (for ISTYLE = 1)

$_{1}$NIELE(1)$_{4}$ - the first element added to the coefficient matrix

$_{5}$NIELE(2)$_{8}$ - the second element added to the coefficient matrix

. .

. .

. .

NIELE(NUMEL1) - the last element added to the coefficient matrix

Card 10 Format (7I5)

Card 10 is identical in nature to card 2 except that its information applies to the construction of the second mesh which is used in the viscous flow solution. Input parameters now have the form "Var 2", e.g., NUMEL1 read in the first mesh is now read in as NUMEL2 for the second mesh. In this case the default values for NNPE2, NVPE2, and NPPE2 become 9, 6, and 3 respectively while no default value for NUMPP2 is specified. The limits for NUMEL2 and NUMVP2 are the same as NUMEL1 and NUMVP1.

Cards 11-15

Cards 11-15 are identical in nature to cards 5-9, respectively, except their information now applies to the second mesh, the mesh used in the viscous flow solution.

NOTE: Included in the FRONT9 output are the parameters JMAX (for the heat transfer mesh) and KMAX (for the viscous flow mesh). The magnitude of these values is currently limited to 6000 for JMAX and 1500 for KMAX in the 7600 or 6600 version. On the NOS version JMAX is limited to 4000 while KMAX is limited to 1000. Increased dimensions of the various stiffness matrices are required for larger problems.

4.4 Input Instructions for ILAST9

Card 1 Format (I10)

Card 1 contains the number of different materials in the meshes

<u>Var</u> <u>Name</u> - <u>Description</u>

$_1$NUMAT$_{10}$ - Number of materials (maximum of [illegible])

Card 2 Format (8E10.3)

Card 2 is read NUMAT times and defines the conductivity and constitutive equations for each of the materials. The conductivity equations are of the form

$$k_x \text{(conductivity in x-direction)} = RKXAI*e^{RKXYA*\theta}$$

$$k_y \text{(conductivity in y-direction)} = RKYAI*e^{RKYA*\theta}$$

while the constitutive equation has the form

$$\tau'_{II} = X_1 e^{X_2/\theta} \epsilon_{II}^{X_3}$$

(For linear analyses $\tau'_{II} = 2X_1 e^{X_2/\theta} \epsilon_{II}$ and $X_3 = 1.0$.) In addition card 2 is used for the calculation of temperature dependent body forces for each of the materials at any given time. The relevant formula is

$$\gamma = \gamma_o(1-\alpha\Delta\theta) \qquad \text{where}$$

$$\Delta\theta = \theta - \theta_{ref} \qquad \text{and}$$

θ_{ref} is the reference temperature and γ is the body forces, also given in this card. Finally, Card 2 allows the user to specify uniform volumetric heat generation in any of the materials.

<u>Var</u> <u>Name</u> - <u>Description</u>

$_1$RHOA(I)$_{10}$ - density of the Ith material

$_{11}$CPA(I)$_{20}$ - heat capacity of the Ith material

$_{21}$RKXA(I)$_{30}$ - RKXA in the conductivity equation for the Ith material

$_{31}$RKXAI(I)$_{40}$ - RKXAI in the conductivity equation for the Ith material

$_{41}$RKYA(I)$_{50}$ - RKYA in the conductivity equation for the Ith material

$_{51}$RKYAI(I)$_{60}$ - RKYAI in the conductivity equation for the Ith material

$_{61}$QQA(I)$_{70}$ - Uniform volumetric heat generation rate for the Ith material

$_{71}$EPSIIA(I)$_{80}$ - minimum strain-rate used in non-linear solution method for the Ith material

$_{1}$X1(I)$_{10}$ - X1 in constitutive equation for the Ith material

$_{11}$X2(I)$_{20}$ - X2 in constitutive equation for the Ith material

$_{21}$X3(I)$_{30}$ - X3 in constitutive equation for the Ith material

$_{31}$X4(I)$_{40}$ - X4 in constitutive equation for the Ith material (not used)

$_{41}$GAMX(I)$_{50}$ - body forces in x-direction for the Ith material

$_{51}$GAMY(I)$_{60}$ - body forces in y-direction for the Ith material

$_{61}$ALPHA(I)$_{70}$ - volumetric expansivity, α, for the Ith material

$_{71}$TREFA(I)$_{80}$ - the reference temperature for the Ith material

Card 3 Format (I10)

Card 3 offers the user the option to release the constraint on certain specified nodal points, that have been constrained by boundary conditions if certain conditions are met.

Var Name - Description

$_{1}$NUMCOD$_{10}$ - the number of nodal points examined for possible release

Card 4 Format (4I5)

If NUMCOD has previously been set to zero, Card 4 should not appear in the data deck. Otherwise Card 4 specifies which nodal points and under what conditions the nodal points are to be released. Card 4 is read as many times as necessary until all NUMCOD boundary conditions are specified.

Var Name - Description

$_1I_5$ - the first nodal point number in a set to have a release constraint

$_6J_{10}$ - the last nodal point number in a set to have a release constraint

$_{11}K_{15}$ - the increment that will be used between nodal point numbers I and J in specifying the same release constraints

$_{10}L_{20}$ - requirement for nodal point release

1 - releases for x-direction positive force

-1 - releases for x-direction negative force

2 - releases for y-direction positive force

-2 - releases for y-direction negative force

Card 5 Format (4F10.4,I10)

Card 5 specifies initial temperatures of the region of analysis, a conversion factor for heat generation, the ambient temperature for surface convection, the surface convection coefficient, and the number of velocity points on each side of an element needed for surface integration of the convection boundary condition.

Var Name - Description

$_1TI_{10}$ - initial temperature of the region

$_{11}FAC_{20}$ - conversion factor for heat generation, 1.0 if units are consistent.

$_{21}TINF_{30}$ - ambient temperature for surface convection

$_{31}CC_{40}$ - convection coefficient (film coefficient)

$_{41}NVPS_{50}$ - number of velocity points on each side of an element (usually 3)

Card 6 Format (I10)

The surface array for convection boundary conditions is initialized to zero, indicating that no convection occurs across any side of any element. Card 6 allows the user to apply convection boundary conditions to any choice of sides on any element. The convention used in numbering the sides of the element results from the element nodal point array. Side 1 is the side consisting of the first three nodal points in the nodal point array. Side 2 is the side with the third, fourth and fifth nodal points and side 3 is comprised of the fifth, sixth and first nodal points.

Var Name - Description

$_{1}NOTZERO_{10}$ - the number of element that are to have convection boundary conditions

Card 7 Format (4I5)

If NOTZERO (Card 6) has been set to 0, then Card 7 must not appear in the card deck. Otherwise, Card 7 contains which elements and what sides the convection boundary condition is applied to. Card 7 is read as many times as necessary until all NOTZERO convection boundary conditions are specified.

Var Name - Description

$_{1}I_{5}$ - the first element number in a set to have a convection boundary condition

$_6J_{10}$ - the last element number in a set to have a convection boundary condition

$_{11}K_{15}$ - the increment that will be used between elements I and J in specifying the same convection boundary condition

$_{16}L_{20}$ - the sides where the convection boundary condition is applied to in element I through J by increments of K

= 1 side 1 only

= 2 side 2 only

= 3 side 3 only

= 4 sides 1 and 2

= 5 sides 2 and 3

= 6 sides 3 and 1

Card 8 Format (E10.3)

Card 8 specifies the decay constant XK1 of the heat sources in the term $e^{XK1 \cdots}$ used in card 10. Enter zero for sources that do not decay.

<u>Var</u> <u>Name</u> - <u>Description</u>

$_1XK1_{10}$ - decay constant for heat sources

Card 9 Format (I10)

Card 9 allows the user to specify a variable number of boundary conditions to be applied to any of the nodal points.

<u>Var</u> <u>Name</u> - <u>Description</u>

$_1NUMBC_{10}$ - number of boundary conditions to be specified

Card 10 Format (4I5,3E10.3)

If NUMBC (Card 9) has been set to zero, then Card 10 must not appear in the card deck. Otherwise Card 10 contains the applied forces or imposed velocities and the surface heat flux or imposed temperature corresponding to each nodal point where a constraint is applied. If an x-constraint is placed on a nodal point (L = 2 or 4), then the FXA2 will be treated as an imposed velocity. Otherwise FXA2 will be treated as an applied force. Similarly, for the y-constraint FYA2 will be treated as an imposed velocity for L = 3 or 4. Otherwise FYA will be treated as an applied force in the y-direction. If L is negative FQA2 is treated as an imposed temperature and if L is positive FQA2 is treated as an applied heat source. Analyses are performed using unit thicknesses for plane strain geometry and one radian section for axisymmetric geometry. Applied forces in axisymmetric analyses are based on the resultant forces from integrating over a section with a one radian arc length. Point sources located at the centerline of axisymmetric regions should be entered as $\frac{1}{2\pi}$ of the total sources.

Var Name - Description

$_1I_5$ - the first nodal point number in a set that has a boundary condition

$_6J_{10}$ - the last nodal point number in a set that has a boundary condition

$_{11}K_{15}$ - the increment that will be used between nodal points I and J in specifying the same boundary conditions

$_{16}L_{20}$ - the type of boundary condition applied to nodal point I through J in increments of K

1 - no constraint on velocity

2 - X constraint on velocity

3 - Y constraint on velocity

4 - X and Y constraint on velocity

if L is negative a specified temperature will also be imposed

$_{21}FXA2_{30}$ - the imposed velocity or applied force* in the x-direction

$_{31}FYA2_{40}$ - the imposed velocity or applied force* in the y-direction

$_{41}FQA2_{50}$ - the imposed temperature or applied heat source

Card 11 Format (2I10)

Card 11 offers the user control of how often data will be sent to the print and tape 8 files from the coupled temperature and flow solutions. The tape 8 file is used in both restarting the current problem and for plotting. Increments refer to the number of time steps in a transient solution or the number of combined viscous flow and temperature solutions in steady state analyses.

Var Name - Description

$_{1}INCPR_{10}$ - increments between prints of the temperature and velocity solutions

$_{11}INCPU_{20}$ - increments between WRITES to TAPE(8) of the temperature and velocity solutions

Card 12 Format (2I10)

When solving nonlinear viscous equations, which requires iteration, Card 12 offers the user control of how often data will be sent to his print and tape 8 files during the iterating procedure. As previously mentioned, tape 8 is used in both restarting the current

*See Appendix A for method of determining nodal point forces from applied pressures.

problem and for plotting. Iterations refer to the number of solution steps used during a nonlinear viscous flow analysis.

Var Name - Description

${}_{1}\mathrm{INTPR}_{10}$ - iterations between prints while solving a nonlinear viscous problem

${}_{11}\mathrm{INTPU}_{20}$ - iterations between writes to tape (8) while solving a nonlinear viscous problem

Card 13   Format (2I10,F10.0)

Card 13 offers the user choices on viscous flow or temperature solutions, coupling or no coupling between the solutions, and transient or steady state analysis. If no coupling is specified, then only viscous flow or temperature analyses are performed, but not both. If coupling is specified, then both viscous flow and temperature solutions are obtained.

Var Name - Description

${}_{1}\mathrm{ITV}_{10}$ - type of solution performed first

1 - viscous solution

-1 - temperature solution

${}_{11}\mathrm{MOP}_{20}$ - coupling between solutions

1 - NO

-1 - YES

${}_{21}\mathrm{ST}_{30}$ - type of analysis

1.0 - transient

0.0 - steady state

Card 14 Format (8I10)

Card 14 offers the user choices on geometry for the problem, methods of solution, reference frames, and the type of matrix solution for the heat transfer problem. In addition it allows the user to specify the maximum number of increments used to attain a solution, the number of iterations for iterative methods (or load increments in incremental solutions), the number of iterations between updates of the stiffness matrix, and whether a linear solution is desired only.

Var Name - Description

$_{1}$ICODE$_{10}$ - geometry

0 - plane strain

1 - axisymmetric

$_{11}$ISOLV$_{20}$ - method of solution of nonlinear viscous flow analyses

1 - secant modulus

2 - initial stress with secant modulus

-2 - initial stress with tangent modulus

3 - tangent modulus

$_{21}$IREFFR$_{30}$ - reference frame

0 - Lagrangian

1 - Eulerian

$_{31}$IHTSOL$_{40}$ - type of matrix solution in heat transfer problem

1 - reformulation of matrix at each time step

2 - LDU decomposition used after first time step

$_{41}$MNI$_{50}$ - maximum number of increments for problem solution

$_{51}$ITMAX$_{60}$ - maximum number of iterations for nonlinear viscous flow solutions

$_{61}$ITCH$_{70}$ - number of iterations between updates of stiffness matrix

$_{71}$LINEAR$_{80}$ - choice of linear solution

0 - nonlinear solutions procedure needed

1 - linear solution procedure needed

Card 15 Format (5E10.3)

Card 15 specifies the maximum problem time to be achieved, the maximum allowed time step to be taken, the maximum allowed displacement during any given time step, the maximum allowed change in the maximal velocity between any given time step, and the maximum allowed change in the maximal strain rate in any given time step.

Var Name - Description

$_{1}$TIMEM$_{10}$ - maximum problem time

$_{11}$DUMAX$_{20}$ - maximum allowed displacement during any time step

$_{21}$DTMAX$_{30}$ - maximum time step

$_{31}$EIIMIN$_{40}$ - maximum allowed change in the maximal strain rate for convergence of nonlinear viscous problems

$_{41}$VMIN$_{50}$ - maximum allowed change in the maximal velocity for convergence of steady-state problems

Card 16 Format (I10)

Card 16 allows the user the choice of printing the TAPE1 data file as part of his output.

Var Name - Description

$_{1}$IWRITE$_{10}$ - print control for the TAPE1 data

EQ. 6 - writes TAPE1 data

NE. 6 - does not write TAPE1 data

Card 17 Format (2I10)

Card 17 allows the user to continue a problem from a previous run that was terminated early.

Var Name - Description

${}_{1}$IDATA${}_{10}$ - continuation of a problem from a previous run:

0 - NO

1 - YES

${}_{11}$NINCR${}_{20}$ - the number of increments to be read from TAPE8 before restarting the analyses. NINCR refers to the net number of increments written onto TAPE8 and not the last increment number that was written.

5.5 Input Instructions for PLOT9

Card 1   Format (I10)

Card 1 allows the user the option to either read in the generated data from MESH9 as cards in his data deck or from a disk file.

<u>Var</u> <u>Name</u> - <u>Description</u>

$_{1}$NTAPE$_{10}$ - the unit number the data from MESH9 will be read in from

5 - cards in a data deck

9 - TAPE9 which is a disk file

Card 2   Format (2I10)

Card 2 specifies information on the number of plots desired and whether a plot consisting of the area within the mesh is required or if symmetry exists about either axis so that not only the area within the mesh is plotted, but also its symmetric half is plotted.

<u>Var</u> <u>Name</u> - <u>Description</u>

$_{1}$NUMPLT$_{10}$ - total number of plots desired (maximum of 100)

$_{11}$ICODE$_{20}$ - type of plot desired

0 - mesh only

1 - symmetry about Y axis

2 - symmetry about X axis

Card 3   Format (20I4)

Card 3 specifies the increment numbers in ascending order that are to be plotted.

Var Name - Description

${}_{1}$ICPL(1)$_{4}$ - first increment number to be plotted

${}_{5}$ICPL(2)$_{8}$ - second increment number to be plotted

.
.
.

ICPL(NUMPLT) - last increment number to be plotted

Card 4 Format (5F10.4)

Card 4 provides dimensions for the plotted data. The bounds of the region provided may correspond to the size of the entire region of analysis or may be some subset of the entire region. Card 4 also allows for scaling of the plotted data.

Var Name - Description

${}_{1}$XMIN$_{10}$ - minimum X value to be plotted

${}_{11}$XMAX$_{20}$ - maximum X value to be plotted

${}_{21}$YMIN$_{30}$ - minimum Y value to be plotted

${}_{31}$YMAX$_{40}$ - maximum Y value to be plotted

${}_{41}$SCALE$_{50}$ - scale factor for reducing or enlarging the plotted data

Card 5 Format (I10,2F10.3)

Card 5 contains the nodal point coordinates which are usually generated by MESH9. If the user has selected the option to read this data from disk, i.e., choosing NTAPE = 9, then these cards are not contained in his card deck.

Var Name - Description

$_{1}I_{10}$ - nodal point number

$_{11}XORD(I)_{20}$ - X-coordinate of nodal point I

$_{21}YORD(I)_{30}$ - Y-coordinate of nodal point I

Card 6 Format (10I7)

Card 6 contains the nodal point numbers and the pressure point numbers that comprise each of the elements. This data is again usually generated by MESH9 and if the user has selected the option to read this data from disk, i.e., choosing NTAPE = 9, then these cards are not contained in his card deck.

Var Name - Description

$_{1}I1_{11}$ - element number

$_{8}NP(I1,1)_{14}$ - first nodal point number comprising the I1 element

$_{15}NP(I1,2)_{21}$ - second nodal point number comprising the I1 element

.
.
.

$_{43}NP(I1,6)_{49}$ - sixth nodal point number comprising the I1 element

$_{50}NP(I1,7)_{56}$ - first pressure point number comprising the I1 element

$_{57}NP(I1,8)_{63}$ - second pressure point number comprising the I1 element

$_{64}NP(I1,9)_{70}$ - third pressure point number comprising the I1 element

Card 7 Format (I10)

Card 7 specifies the number of boundary points that are to be plotted.

Var Name - Description

$_{1}NUMBPT_{10}$ - the number of boundary points to be plotted (maximum of 100)

Card 8 Format (3I5)

Card 8 specifies the numbers of the nodal points that are to be used as boundary points in plotting. Card 8 is read as many times as necessary until all NUMBPT are specified.

Var Name - Description

$_{1}I_{5}$ - the first nodal point number in a set used as a boundary point

$_{6}J_{10}$ - the last nodal point number in a set used as a boundary point

$_{11}K_{15}$ - the increment that will be used between nodal point I and J in specifying the boundary points

Card 9 Format (3F10.4)

Card 9 specifies the vector length in the vector plots, the increment between isotherms in the plots, and the lower limit for plotting velocities (i.e., velocities that are insignificant in comparison to most velocities may be deleted from the plots).

Var Name - Description

$_{1}VECTL_{10}$ - vector length in vector plots

$_{11}CTEMP_{20}$ - increment between isotherms in the temperature plots

$_{21}CUTOFF_{30}$ - the lower limit for plotting velocities

Chapter V

CONCLUDING REMARKS

This document is intended to provide illustrative problems that are to be used as examples by the user in setting up his own problems for solution by COUPLEFLO. In addition, control streams and input instruction to COUPLEFLO are contained. A general description of the governing equations and finite element formulations that form the basis of COUPLEFLO are found in Part I-Theoretical Background.

REFERENCES

1. P. R. Dawson and P. F. Chavez, COUPLEFLO--A computer Code for Coupled Creeping Viscous Flow and Conductive-Convective Heat Transfer--Part I Theoretical Background, SAND78-1406.